ON CONTACT MODELLING IN ISOGEOMETRIC ANALYSIS FOR VEHICLE SAFETY

Submitted in partial fulfilment of the requirements for the award of the degree of

MASTER OF TECHNOLOGY

IN

Transportation systems

(With specialization in Infrastructure systems)

BY

ANUJ KUMAR

(17554004)

Under Guidance of

Dr. A. K. SWAIN

Assistant Professor

IIT Roorkee

CENTRE FOR TRANSPORTATION SYSTEMS

INDIAN INSTITUTE OF TECHNOLOGY, ROORKEE

ROORKEE-247667

CANDIDATE'S DECLARATION

I hereby declare that the work carried out in the dissertation entitled "**ON CONTACT MODELLING IN ISOGEOMETRIC ANALYSIS FOR VEHICLE SAFETY**" is presented on behalf of partial fulfilment of the requirement for the award of the degree of Master of Technology with specialization in **Infrastructure systems** submitted to the centre for Transportation systems (CTRANS), Indian Institute of Technology Roorkee, India, under the supervision and guidance of **Dr. Abinash Kumar Swain**, Assistant Professor MIED and associate faculty CTRANS, IIT Roorkee, India. I have not submitted the matter embodied in this report for the award of any other degree or diploma.

Date- 28/06/2019

Place -Roorkee

Anuj Kumar

Enroll no. 17554004

M. Tech, Infrastructure syatems

CTRANS, IIT Rorrkee

CERTIFICATION

This is to certify that the above statement made by the candidate is correct to the best of our knowledge and belief.

Dr. Abinash Kumar Swain

Assistant Professor,

MIED, IIT Roorkee

UK-247667, India

ACKNOWLEDGEMENT

I wish to express my deep sense of gratitude and sincere thanks to **Dr. Abinash Kumar Swain** assistant professor MIED and associate faculty in **Centre for Transportation systems (CTRANS),** IIT Roorkee, for being helpful and a great source of inspiration. I would like to thank him for providing me with an opportunity to work on this excellent and innovative field of research. His keen interest and constant encouragement gave me the confidence to complete my work. I wish to thank him for his constant guidance and suggestions without which I could not have successfully completed this thesis. I would also like to thank Dr. Durga Toshniwal, Head of Center CTRANS, IIT Roorkee for his constant support during my study. Also, I would like to thank all the teaching and non-teaching staff members of the MIED and CTRANS who have contributed directly or indirectly in successful completion of my seminar work.

I am very thankful to my parents & all of my friends for their never-ending encouragement in bringing out this seminar report.

Date - 28/06/2019

Place- Roorkee

Anuj Kumar

M. Tech, Infrastructure systems

Enrolment No:17554004

CTRANS, IIT Roorkee

ABSTRACT

This work addresses the study on contact modelling in isogeometric analysis. Research papers including vehicle crashworthiness and occupant protection, on contact modelling in isogeometric analysis, contact searching algorithm is described. CAD modelling of front vehicle bumper and rigid pole is done after that model is discretized using software. Isogeometric analysis has been described as a tool for geometric modelling and surface discretization. MATLAB software package is used for generating mesh, applying the boundary condition, solving the system of equations for stress and displacements and visualising the solution results. For comparison of the results between FEM and Isogeometric analysis an ABAQUS/CAE simulation is used.

TABLE OF CONTENTS

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Background

India is the sixth largest in terms of the number of vehicles in the world and one of the world's second largest road network. But equally staggering is its 1.46 lakh fatalities arising from more than 5 lakh road accidents every year. In excess of the shattering loss of life and trauma to the victims and their families, economists say that India suffers an estimated 3% drop in GDP from this every year. That brings us to a pressing question – are our roads and more importantly, our cars safe enough? Vehicle safety plays an important role in reducing crash fatalities. There are numbers of rules and regulations are made for vehicle safety and if they are applied, they can save many a life. These regulations include manufacturers to meet with sufficient vehicle strength in front, side and rear impacts. These also includes stability and control of vehicle with assurance of airbags and seatbelts are fitted in all vehicles.

Analysing this, safety is paramount importance in designing and manufacturing of vehicle. Crashworthiness is the first step to be completed in vehicle design.

1.2 Vehicle crashworthiness:

This is the ability of a vehicle structure to protect its occupants going through an impact crash [1]. This is not limited to vehicle crashes; it is also applicable to other transportation sectors like planes ships and rails. Between 1879 to 1890[2] the first systematic and successful investigation of crashworthiness was implemented in railway axel. In other words, it is the process of improving crash performance vehicle structure by going through an impact crash [3]. For improving the structural design for crashworthiness, we need to understand the crash characteristics.

1.3 Crash Characteristics:

i. Displacement and energy: In modern design style of vehicle frontal structural length is being reduced and it is ensured that this design structure should absorb most of the impact energy to minimize its effect up to the occupants.

ii. Crash pulse: When vehicle go through an impact crash, suddenly velocity becomes zero due to inertia, human body feels deceleration to its velocity this is called crash pulse. For the measurement of crash pulse to brain damage head injury criterion (HIC) is used and it should be in certain limit by regulations [4].

iii. Crash position: The structure should be designed in such a way that it should overcome from any type or in any crash positions like full frontal impact, side impact, offset frontal impact and rollover.

iv. Automobile compatibility: Automobile should be designed in such a way that its structure should be able to mitigate the injuries from an accident with two different automobiles having different weight and size.

1.4 Types of motor vehicle collisions:

i. Frontal impact
ii. Rear impact
iii. Lateral impact
iv. Rotational impact
v. Rollover

1.5 Modelling Vehicle Structures:

i. Lumped Mass Spring (LMS) model
ii. Finite Element (FE) model
iii. Multi Body Dynamics (MBD) model
iv. Hybrid model

I. LMS Model:

This model was developed by Kamal [5] in 1970. This is simple model with relatively greater accuracy. In was known as Lumped Mass-spring model because analysis of the vehicle components is assumed to be a system of discrete masses and springs. Finding the spring and mass characteristics is the main objective of this model. This model was successfully used in full automobile simulation with rigid wall. The following figure is an approximation of vehicle by system of lumped masses and springs.

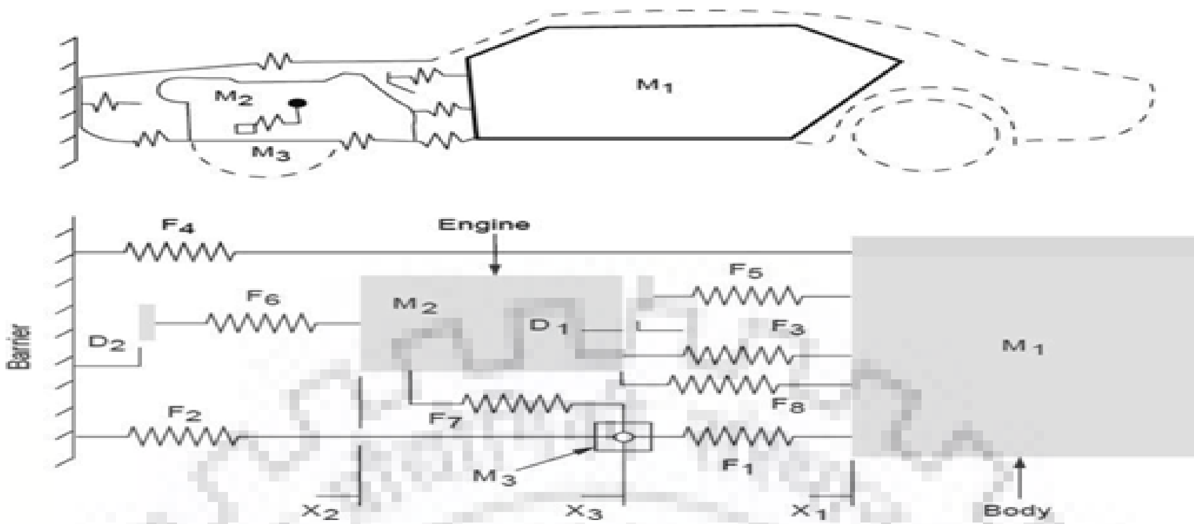

Figure 1.1: LMS Model [5]

II. Finite Element (FE) model:

In the finite element vehicle crash model full and detailed geometric description of vehicle components and their material properties are utilized for analysis. For the finite element modelling the whole vehicle body is divided into primitive finite elements [6]. The elements may be chosen 1-D, 2-D or 3-D according to the type of analysis. 1. Linear elements (1-D) 2. Triangular elements (2-D) 3. Rectangular elements (2-D) 4. Brick elements (3-D). The Finite element analysis can be done in FE software packages like LS-DYNA, ANSYS, INVSYS, MATLAB, MAPLE and MATHMATICA. The CAD model of vehicles are imported to the software. Using the mesh tool, the finite element meshes are generated in the vehicle body. On giving the forces or deformation input to the software as well as the material specification it simulates the vehicle crash and give the stress and deformation for all the elements.

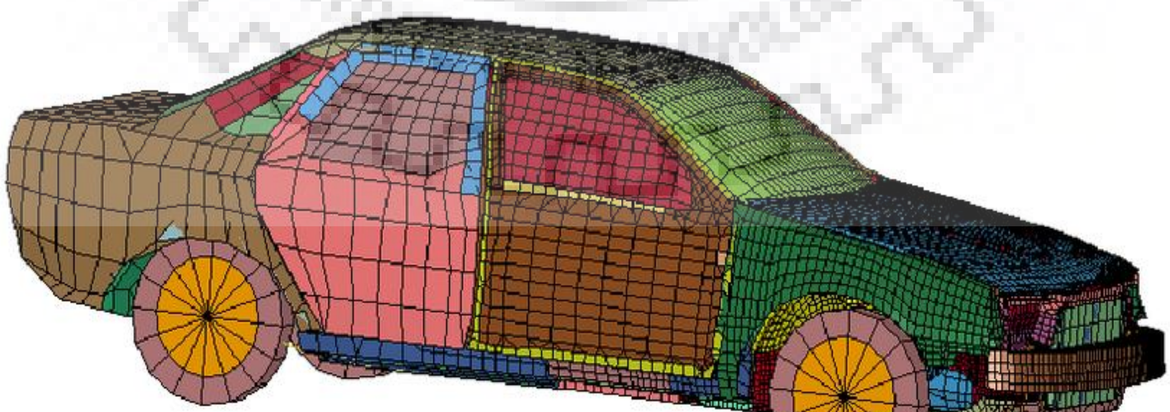

Figure 1.2: Finite Element (FE) model [6]

III. Multi Body Dynamics (MBD) model:

LMS model is special case of MBD model. In this model physical component of human body is represented by number of inter-connected bodies having different joints. The joints can be of different types and can have different degrees of freedom. The inter-connected bodies can be flexible or rigid but in LMS model they must rigid.

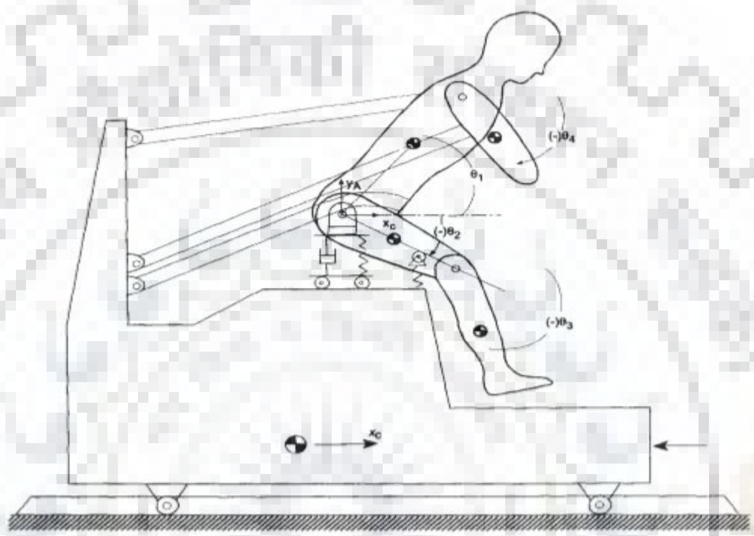

Figure 1.3: Multi Body Dynamics (MBD) model [7]

IV. Hybrid model:

The Hybrid model [7] is the combination of MBD and LMS models. As we know MBD is used to simulate occupant body where FE is used for full vehicle structural simulation. This model has greater efficiency because in MBD model computational cost is low compared to FE model. The following figure shows the hybrid model [7] for a side impact test. LS-DYNA is used for modelling of vehicle structure and rigid barrier, where MADYMO (an MBD commercial software) is used to model the occupant.

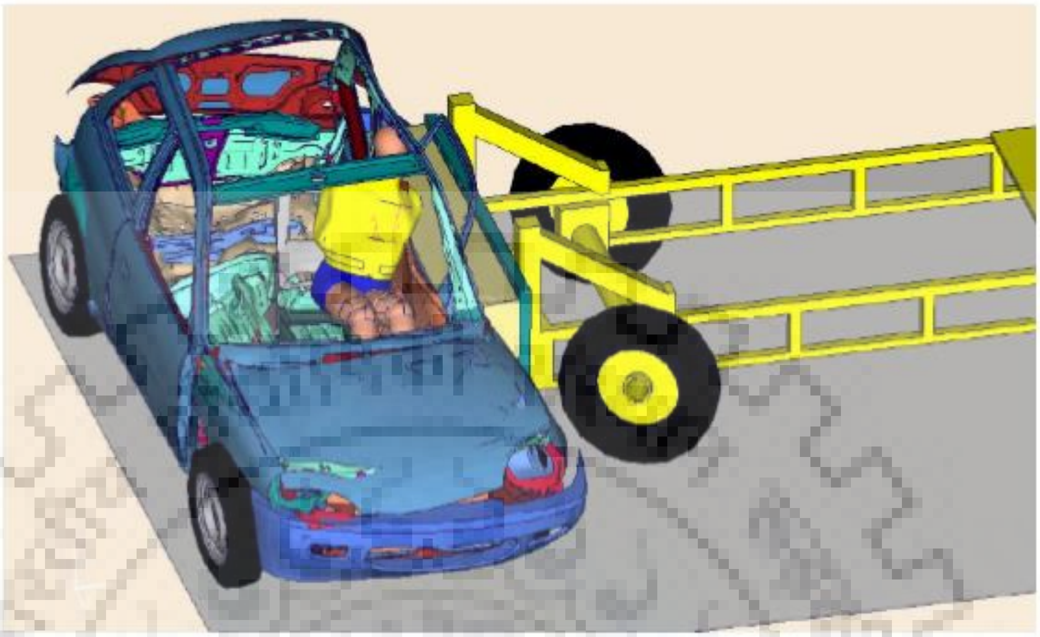

Figure 1.4: Hybrid model [7]

1.6 Objective:

The aim of this work is modelling of contact between rigid pole and deformable bumper during low velocity impact using Isogeometric analysis and to compare it with conventional finite element method.

1.7 Scope of thesis:

- To develop impact contact simulation procedure for front vehicle bumper with rigid pole using FEM software like ABAQUS.
- To formulate same contact modelling in Isogeometric analysis using MATLAB and RHINOCEROUS.
- To compare the results of both, contact modelling procedures and show that which is better for designing of vehicle bumper for safety purpose.

CHAPTER 2

LITERATURE REVIEW

In this chapter some of researcher's works are summarised. Publications related to vehicle crashworthiness, vehicle crash models and their comparative studies are given in brief. Contact searching algorithms and other relevant works are presented briefly.

2.1 Vehicle crashworthiness design:

GJ Gao and HQ Tian [14] published an article on vehicle's crashworthiness design and collision analysis. In this paper vehicle crashworthiness, types of collisions are described. This is an American criterion which presents the evaluation criteria of the middle and end part of the vehicle. These are values of the absorbing energy of the middle and end of the vehicle and acceleration of the cabin. C.M. Ni and J.O. Song: [15], they published computer aided design analysis methods for vehicle structural crashworthiness, symposium on vehicle crashworthiness including impact biomechanics. In their model use of FEM for finding out stresses and deformation at critical section is important.

2.2 Vehicle crash models and their parameters:

P. Jonsen et al. [16], in this paper they studied frontal crash and compared the obtained results with vehicle crash data. They used INVSYS software to identify the parameters of vehicle crash model. FEM and LPM both have been used for the analysis. P. D. Bios wrote a book entitled Vehicle Crashworthiness and Occupant Protection in Transportation System. In this book FEM and LPM has been described along with their comparison. Yehia A. Abdel-Nasser [8] published Frontal crash simulation of vehicles against lighting columns using FEM. In this paper study of vehicle impact with lighting column is described and all the formulation is based on finite element method. Effect of impact forces on different part of vehicle structures is calculated and plotted as stress and deformation.

2.3 Contact searching algorithms:

Wang Fujunet al. [26] presented a paper called Contact searching algorithm for contact impact problem. Detailed description of all three methods of contact searching algorithms is given in this paper.R. P. R. Cardoso & O. B. Adetoro [10] published a paper On contact modelling in isogeometric analysis. In this paper they presented contact formulation of sheet metal forming in isogeometric analysis. Die tool and sheet metal both are discretised using NURBS as basis function contact constraints are measured and boundary conditions are applied. Result is compared with finite element contact formulation and it is shown that NURBS contact formulation is better than conventional FEM in every aspect. Song Wang, and Akitake Makinouchi [12], in his work contact searching and modelling is presented for blow moulding process. They proposed the implementation of three kind of global search algorithms and compared the result with two existing methods in terms of accuracy and CPU time.

2.4 Other relevant work:

Vinh Phu Nguyenet al. [11] presented an analysis for solid and structural mechanics using MATLAB and implementing it for one, two or three dimensional solid and structures in isogeometric analysis. Toolbox with codes is described in detailed for fracture and impact problems. Zefeng Wen et al. [18] In this paper ANSYS/DYNA is used for contact simulation between wheel-rail contact-impact. The effect of axle load and train speed on contact forces, the stresses and strains in the railhead are investigated in detail. Daniel Barbedo Vasconcelos Santosa Alex Alves Bandeira [19] published an article on numerical modelling of contact problems with the finite element method utilizing a B-Spline surface for contact surface smoothing. In this article the problems exhibiting large displacements, curved contact surfaces, as well as large sliding were presented. In all cases presented, convergence was attained, and the displacements obtained were consistent with the results generated in ANSYS. Temizer et al. [24], in this paper they numerically presented the behaviour of NURBS based isogeometric analysis in contact problem and compared the result with C^0-continuous LaGrange finite element method.

CHAPTER 3

ISOGEOMETRIC ANALYSIS: INTRODUCTION

Design and analysis are two main aspects of mechanical structures either static or dynamic. Design is done in CAD environment and then sent to some finite element computational environment for analysis. FEM converts the original smooth CAD model to non-smooth finite element meshes. Geometric approximations during changing the model from CAD to FEM increase error in analysis results. Isogeometric analysis introduced by T.R.J. Hughes [27] solves this problem which is an analysis method which may be applied directly to the smooth CAD model. Computer aided design environments use NURBS to interpolate between the control points and NURBS consist of inbuilt elements in between their knots. These elements create mesh on smooth geometry without approximating it. It is more powerful method for design and analysis compared to FEM. The main idea to develop a computer aided design model in such a way that it can be used for the analysis purpose also first given by Hughes in 2005. NURBS are capable of representing the complex geometries by its versatile basis functions. Later found in this field of research that NURBS face difficulty in local refinements. Another very powerful technique for geometric representation is T-Spline. It is much more versatile than NURBS. It is a more general form of NURBS which enables local refinement strategies to work efficiently

3.1 NURBS (Nonuniform Rational B-spline):

NURBS is used for geometric modelling of different objects and structures. NURBS has been used extensively in CAD softwares as basis for modelling. NURBS has following properties and features

3.11 Knot vectors:

In this contact modelling open knot vector will be used. These are the set of non-negative parametric co-ordinates. These are repeated p + 1 time at beginning and the end of the vector, where ‘p’ is degree of the polynomial basis functions. One dimensional basis function having degree ‘p’ is defined by

$$\Xi = \{\ \xi 1, \ldots., \xi 2, \ldots, \xi m + p + 1, \}$$

m = number of control points or basis function.

If basis function degree ‘p’ it will have p-1 continuous derivative

3.12 Control points and basis functions:

The basis function will be obtained recursively by using following formulae

$$N_I^p = \frac{(\xi - \xi 1)}{\xi 1 + p - \xi 1} N_I^{P-1}(\xi) + \frac{\xi I + p + 1 - \xi}{\xi I + P + 1 - \xi 1} N_{I+1}^{p-1}$$

Where ‘ξ’ is local parametric co-ordinate.

P = degree of basis function.

This formula initialised with piecewise polynomial basis function and if the degree p = 0, i.e.:

$$N_I^0 = \{ \begin{matrix} 1 & if\ \xi 1 \leq \xi < \xi I+1 \\ 0 & Otherwise \end{matrix} \}$$

All piecewise polynomial basis function obtained from the above recursive formulae from a partition of unity it means:

$$\sum_{I=1}^{m} N_J^p(\xi) = 1, \quad \xi \in \Xi = [\xi 1, \xi m + p + 1]$$

The basis functions are always positive. i.e. $\not\forall\ \xi \rightarrow N_J^p(\xi) \geq 0$

A B-spline surface is constructed from a net of control points A_{IJ} and a two-dimensional knot set $\Xi\ X$ H with $H = \{\eta_{1,}\ \eta_2 \ldots .\ \eta_{n+q+1}\}$ where q and n is the degree and number of control points . A B-spline surface is given by following linear combinations.

$$S^{p,q}(\xi, \eta) = \sum_{I=1}^{n} \sum_{J}^{m} N_I^q(\eta) N_J^q(\xi) A_{IJ}$$

Solid B-Spline is constructed using a net of control points A_{IJK} and a 3-dimensional set Ξ X H X Z, with Z = $\{\zeta_1\ , \zeta_2 \ldots \ldots , \zeta_{I+n+1}\}$

Where *r* and *l* are the degree and number of control points along the ζ direction. Local approximation is given by following linear combinations.

$$T^{p,q,r}(\xi,\eta,\zeta) = \sum_{k,I,J=1}^{l,n,m} N_K^r(\zeta)N_I^q(\eta)N_J^p(\xi)A_{IJK}$$

NURBS is the weighted linear combination of the basis functions, a NURBS surface is obtained from a 2-dimensional knot set Ξ X H and a net of control points A_{IJ} and weights W_{IJ}.

$$^{w}S^{p,q}(\xi,\eta) = \frac{\sum_{I=1}^{n}\sum_{J}^{m} N_I^q(\eta)N_J^p(\xi)W_{IJ}A_{IJ}}{W}$$

The NURBS solid is constructed using a 3-dimensional knot set Ξ X H X Z and net of control points.

$$^{w}T^{p,q}(\xi,\eta,\zeta) = \frac{\sum_{k,I,j=1}^{n,n,m}\sum_{J}^{m} N_I^q(\eta)N_J^p(\xi)W_{IJ}A_{IJ}}{W}$$

With the NURBS surface

$$W = \sum_{I=1}^{n}\sum_{J}^{m} N_I^q(\eta)N_J^q(\xi)W_{IJ}$$

And with NURBS solid

$$W = \sum_{k=1}^{l}\sum_{I=1}^{n}\sum_{J=1}^{m} N_K^r(\zeta)N_I^q(\eta)N_J^p(\xi)$$

The NURBS control net and geometric approximation is given by in Figure 3.1

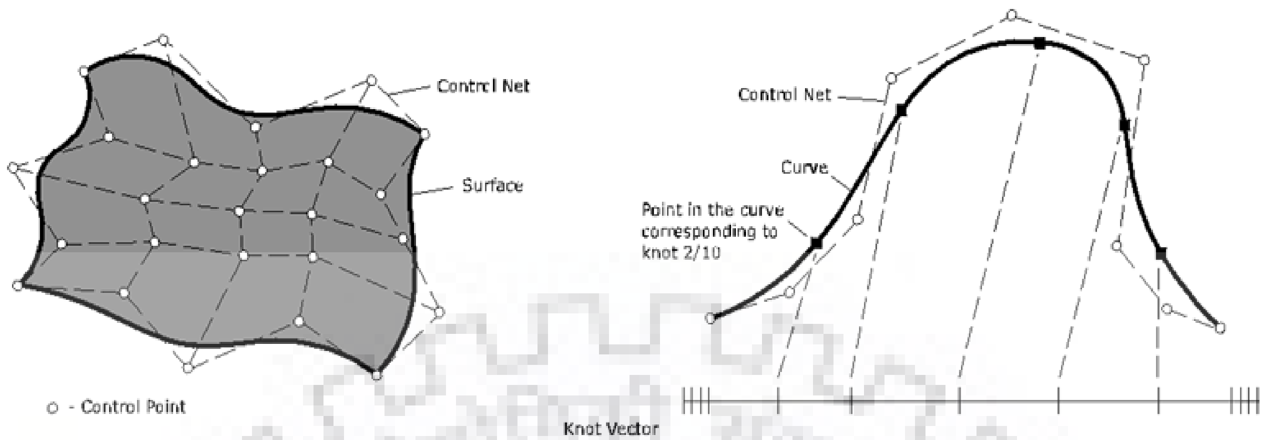

Figure 3.1: NURBS surface, its control net and approximation with basis function [10]

CHAPTER 4

CONTACT DETECTION OR CONTACT SEARCHING ALGORITHMS

After meshing both objects have been discretised into smaller parts. Meshing is done using either finite element mesh or NURBS (isogeometric analysis) mesh. finite element meshing is used in almost every contact modelling because NURBS is a new concept and has not been implemented much. Contact searching mechanics concept is same in both finite element analysis and isogeometric analysis, it has three major steps.

I. Master and slave surfaces:

Contact modelling is done between master and slave surfaces. Rigid body (Pole) is called master surface and deformable (vehicle bumper) is termed as slave body. All the contact searching algorithm will be based on these two surfaces.

II. Global search phase:

This is a first search performed such that potential (which are supposed to hit with each other) contact nodes (nodes are position at corners of rectangular mesh). Global means we will find the approximate contact area having nodes which are supposed to be hitting with each other (master and slave surfaces).

III. Local search step:

In this search we will find exact contacting or hitting nodes (control points in NURBS) of master and slave surfaces which are hitting with other.

IV. Penetration/gap should be zero:

The penetration or gap between two contacting surfaces should be zero means they just should touch each other this can be done making nodal (knot) distance equation equals to zero. All steps will lead to find exact contacting nodes (knot in IGA) after this we will apply boundary conditions to nodes (knots) of both contacting surfaces (master and slave) and result will be displayed in form of stress or deformation plot. For contact modelling (meshing and contact detection) we are using finite element method for a long time which is quite time taking and has less efficiency compared to NURBS. Here I will use NURBS and will compare the result with conventional finite element analysis.

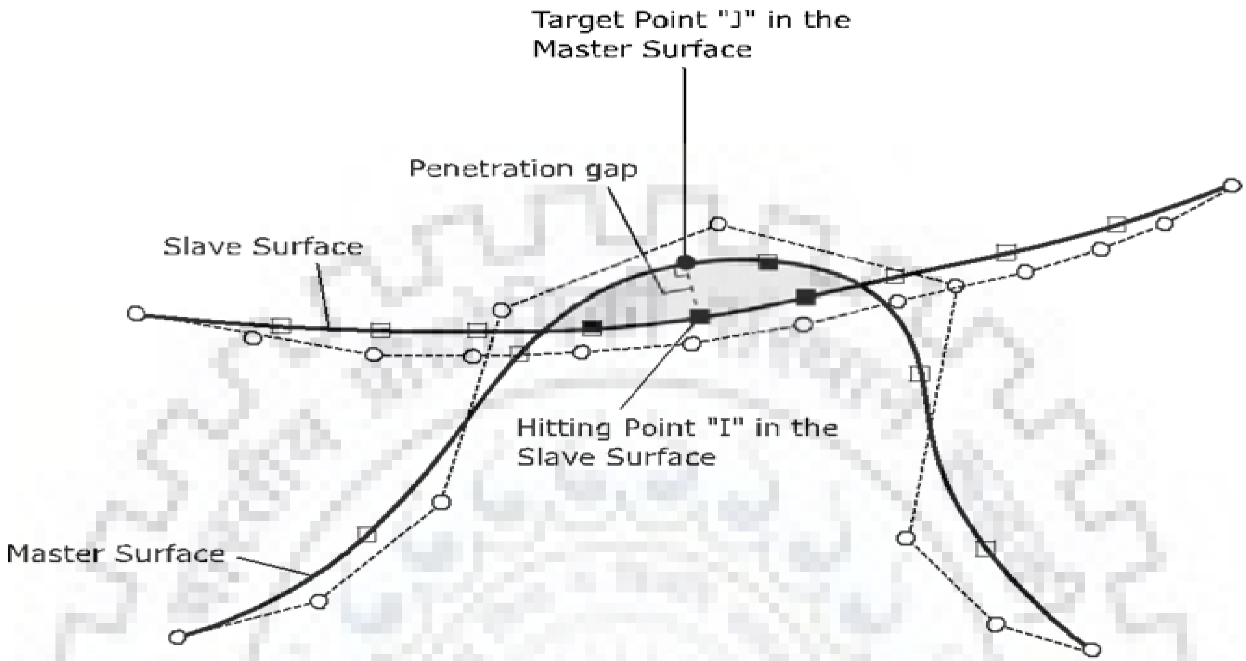

Figure 4.1: Contact detection between master and slave surfaces [10]

Gap function 'g'

gap = – penetration

This function is asymmetric function

Gap function is defined for

- separation $g > 0$
- contact $g = 0$
- penetration $g < 0$

Gap function governs normal contact

Gap function is given by: -

$$g(\xi^{i+1}, \eta^{i+1}) = \{x_h - S(\xi^{i+1}, \eta^{i+1})\}. V_n(\xi^{i+1}, \eta^{i+1})$$

Where $g(\xi^{i+1}, \eta^{i+1})$ is gap function for calculating normal gap.

x_h are the coordinates of the hitting knot.

$g(\xi, \eta)$ is the NURBS contact surface.

$V_n(\xi^{i+1}, \eta^{i+1})$ is the unit normal vector at the contact knot in the target NURBS surface.

In contact mechanics we have to optimise the gap function for zero penetration gap for this purpose we can use Newton-Raphson, Newton iterative or any other optimisation technics [13].

4.1 Contact modelling examples:

There are following contact modelling examples which are performed in various sectors.

- Assembled parts, e.g. engines
- Railroad contacts
- Gears and bearings
- Breaking systems
- Tire-road contact
- Metal forming
- Crash tests
- Biomechanics
- Granular materials
- Electric contacts (Simulation of electric current)
- Tectonic motions (Geography)
- Deep drilling
- Impact and fragmentation

CHAPTER 5

CONTACT MODELLING USING FINITE ELEMENT METHOD (FEM)

Contact modelling in FEM is a main step followed by some prior work like Model construction, Material properties assignment, Meshing, Steps and interaction there is following process chart for FEM contact simulation for impact of vehicle front bumper with rigid pole.

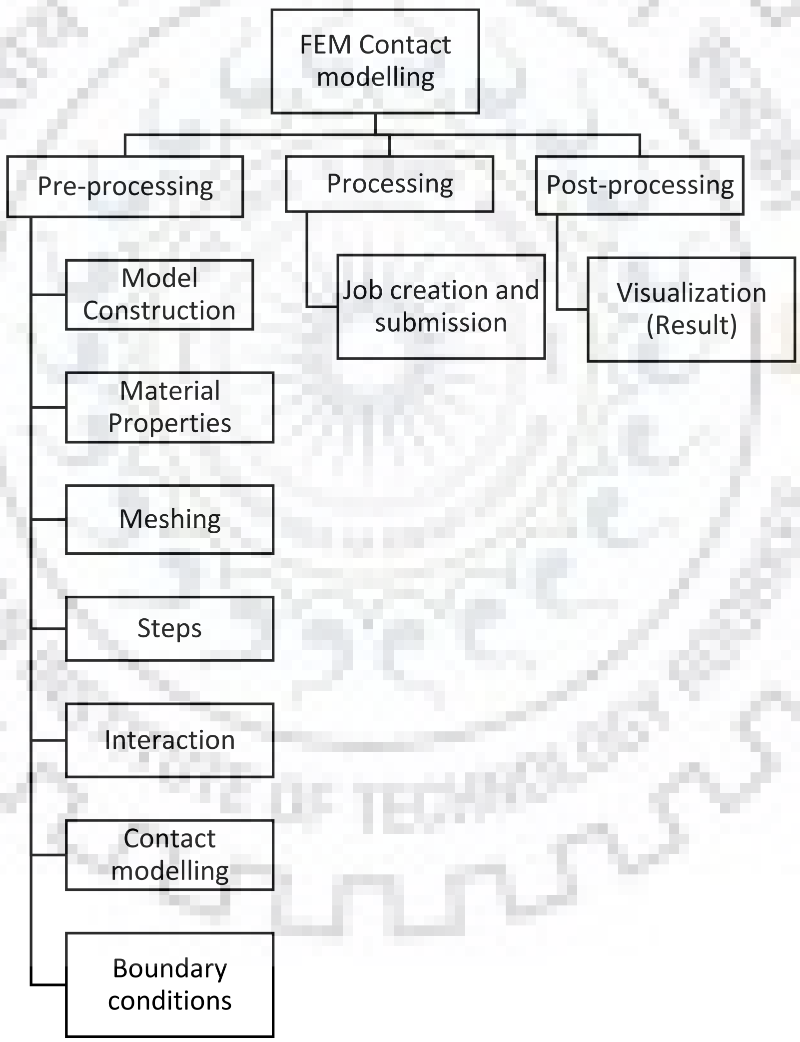

Figure 5.1: FEM simulation procedure

5.1 Pre-processing:

Pre-processing is performed before going for job submission to the processor. Here input is provided to processor it contains following steps.

5.11 Model construction:

CAD Models of Bumper and pole are created in modelling software. There are many modelling softwares are available for CAD geometry like.

- 3DS MAX
- BLENDER
- CATIA
- SKETCHUP
- RHINOCEROS
- SOLIDWORKS
- AUTOCAD
- INVENTOR

Here in this simulation I have used SOLIDWORKS for modelling of pole and bumper and geometry is saved as ‘**.iges**’ file.

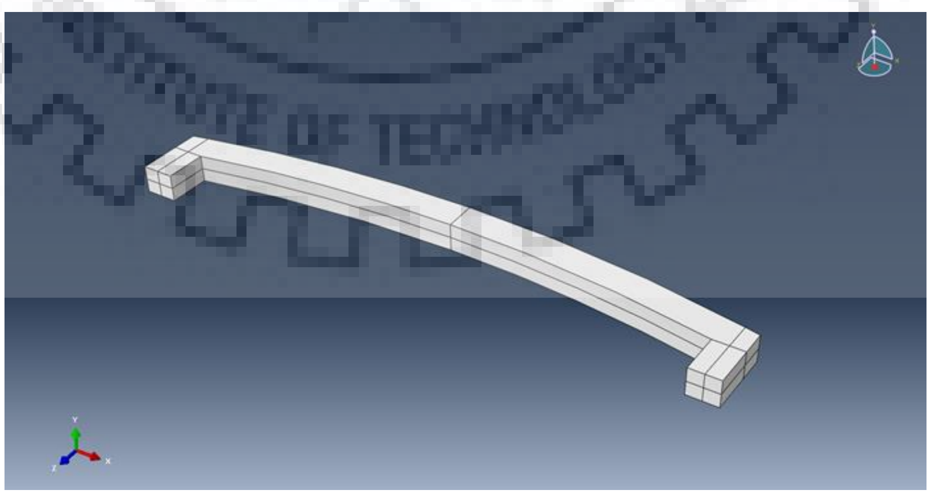

Figure 5.2: Bumper model

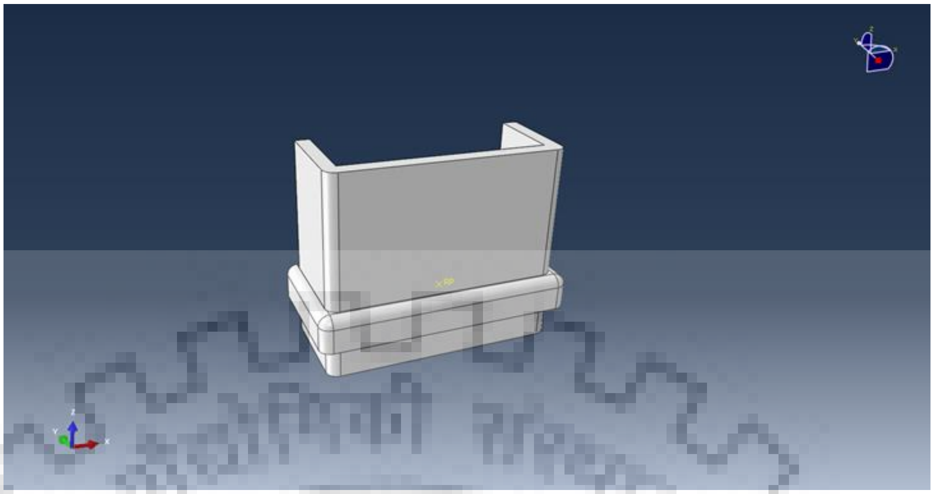

Figure 5.3: Pole model

After creating model in SOLIDWORKS it is imported in ABAQUS

Material properties assigned to the bumper are Young's modulus = 210 GPa, Poisson's ratio = 0.28. Since the created model is solid homogeneous so the properties will be assigned to whole geometry instead of upper shell. For **Pole** model no properties are assigned as it is kept as **discrete rigid** as our aim is only bumper so we don't want to pole to be deformed and it generally has much strength compared to bumper so there will be no effect on pole during analysis. Both the model (rigid pole and deformable bumper) are assembled and kept such that it just touching each other.

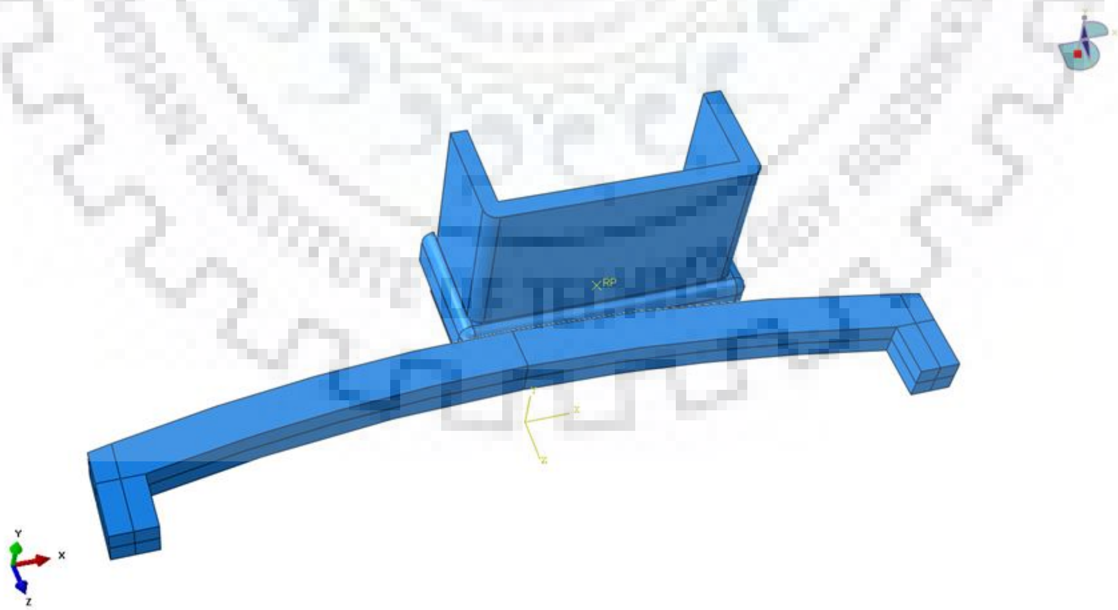

Figure 5.4: Bumper and Rigid pole assembly

5.12 Meshing:

Meshing is defined as dividing the whole model into smaller elements in other words it is nothing but discretization of continuous body into finite number of elements. It is done to ensure proper distribution of loads over whole body. Meshing affects the result a lot. Mesh generation can be performed in various softwares like

ANSYS, OPEN FOAM, GMESH, ABAQUS, NATRAM, BLENDER, RHINO, LS-DYNA, SIMSCALE.

In this simulation meshing is performed in Abaqus using ABAQUS mesh module

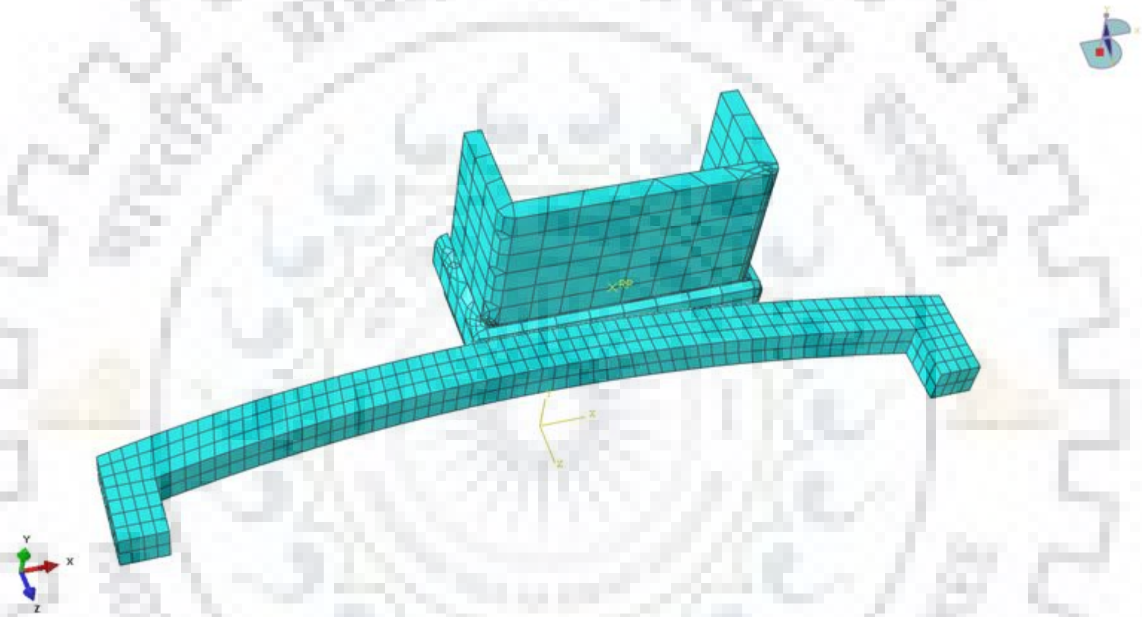

Figure 5.5: Bumper-rigid pole mesh model assembly

Number of elements is	1850
Number of internal elements generated for contact	472
Number of nodes is	2775
Number of nodes defined by the user	1831
Number of internal nodes generated by the program	944
Total number of variables in the model	6912

Here both bumper and pole are meshed using rectangular mesh. Since pole is taken as rigid so course meshing can be used for pole which reduces the total analysis time.

5.13 Steps:

Steps are the unit of the FEM dynamic contact interaction. The total time in simulation will depends upon how many steps are taken place in a simulation. It basically means increment in displacement as bumper moves towards pole so we assign minimum and maximum incremental size of each step in seconds. There is step module in ABAQUS where input data is given and it will be valid for each step.

5.14 Interaction:

After specifying the steps, we have to stablish contact interaction between desired objects here in this simulation the desired surfaces to be interacted are bumper frontal surface and rigid pole surface both are assigned for interaction like this.

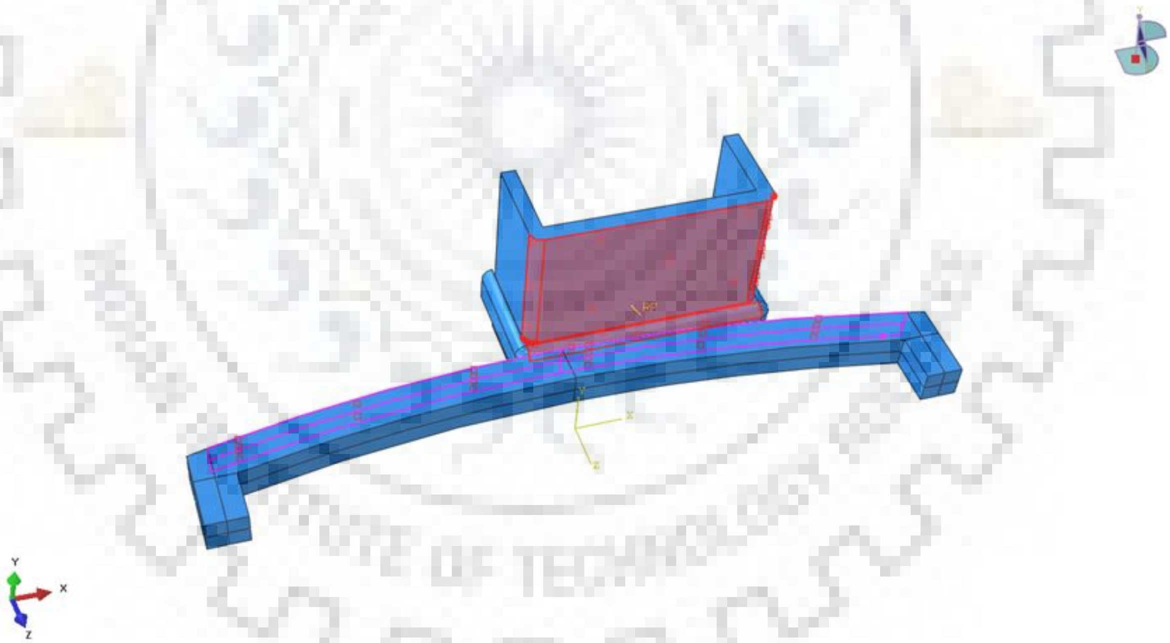

Figure 5.6: Interacting surfaces

As shown in figure 5.5 both surfaces are highlighted, the total number of elements which will come in contact to each other are **472** in numbers.

5.15 Contact modelling:

Because physical contacting bodies do not interpenetrate, the application must establish a relationship between the two surfaces to prevent them from passing through each other in the analysis. When the application prevents interpenetration, it is said to enforce contact compatibility.

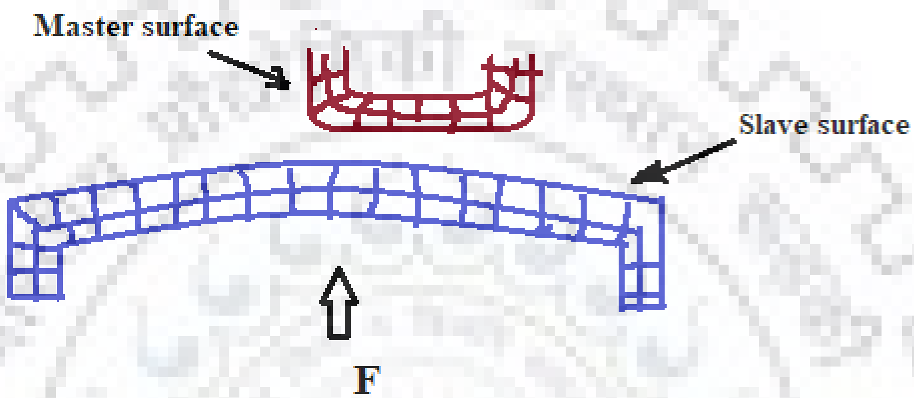

Figure 5.7: Bumper initial position before contact

In order to enforce compatibility at the contact interface we have some optimisation methods like Penalty methods, LaGrange method. In ABAQUS contact simulation, we have used Penalty method to reduce interpenetration master-slave surfaces.

I. Penalty and Lagrange Contact Formulation:

For nonlinear solid body contact of faces, Penalty or Augmented Lagrange formulations can be used. Both of these are penalty-based contact formulations:

$$F_N = k_{Normal} \times X_{Penetration}$$

The finite contact Force, F_N, is a concept of contact stiffness, k_{Normal} . The higher the contact stiffness, the lower the penetration, xp, as illustrated here

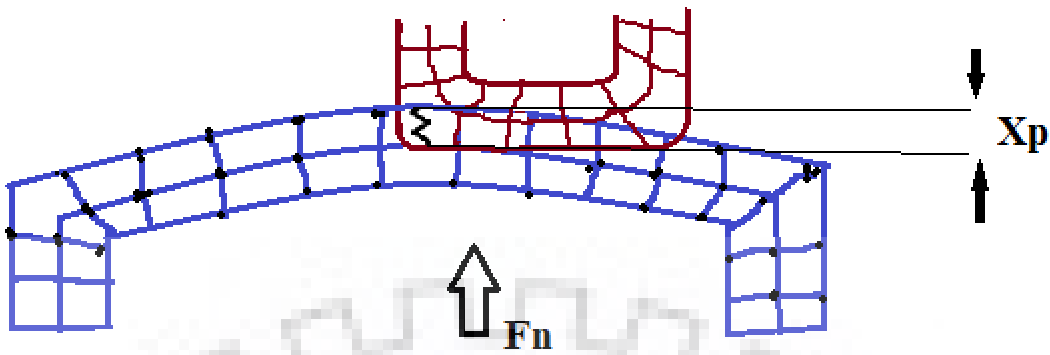

Figure 5.8: Pole penetrates bumper

Ideally, for an infinite k_{Normal} one would get zero penetration. This is not numerically possible with penalty-based methods, but as long as $X_{Penetration}$ is small or negligible, the solution results are accurate. The main difference between Pure Penalty and Augmented Lagrange methods is that Augmented Lagrange augments the contact force (pressure) calculations:

Penalty method: $F_N = k_{Normal} \times X_{Penetration}$

LaGrange multiplier method: $F_N = k_{Normal} \times X_{Penetration} + \lambda$

Because of the extra term λ, the Augmented Lagrange method is less sensitive to the magnitude of the contact stiffness k_{Normal}.

5.16 Boundary Conditions:

Boundary conditions are given to the different elements to get desired degree of freedom and constrained motion for rigid Pole all of the motions are kept constrained so degree of freedom of pole will be Zero and Bumper will have to move in a particular direction with some definite velocity (v = 40 m/s) so that it get impacted with rigid pole

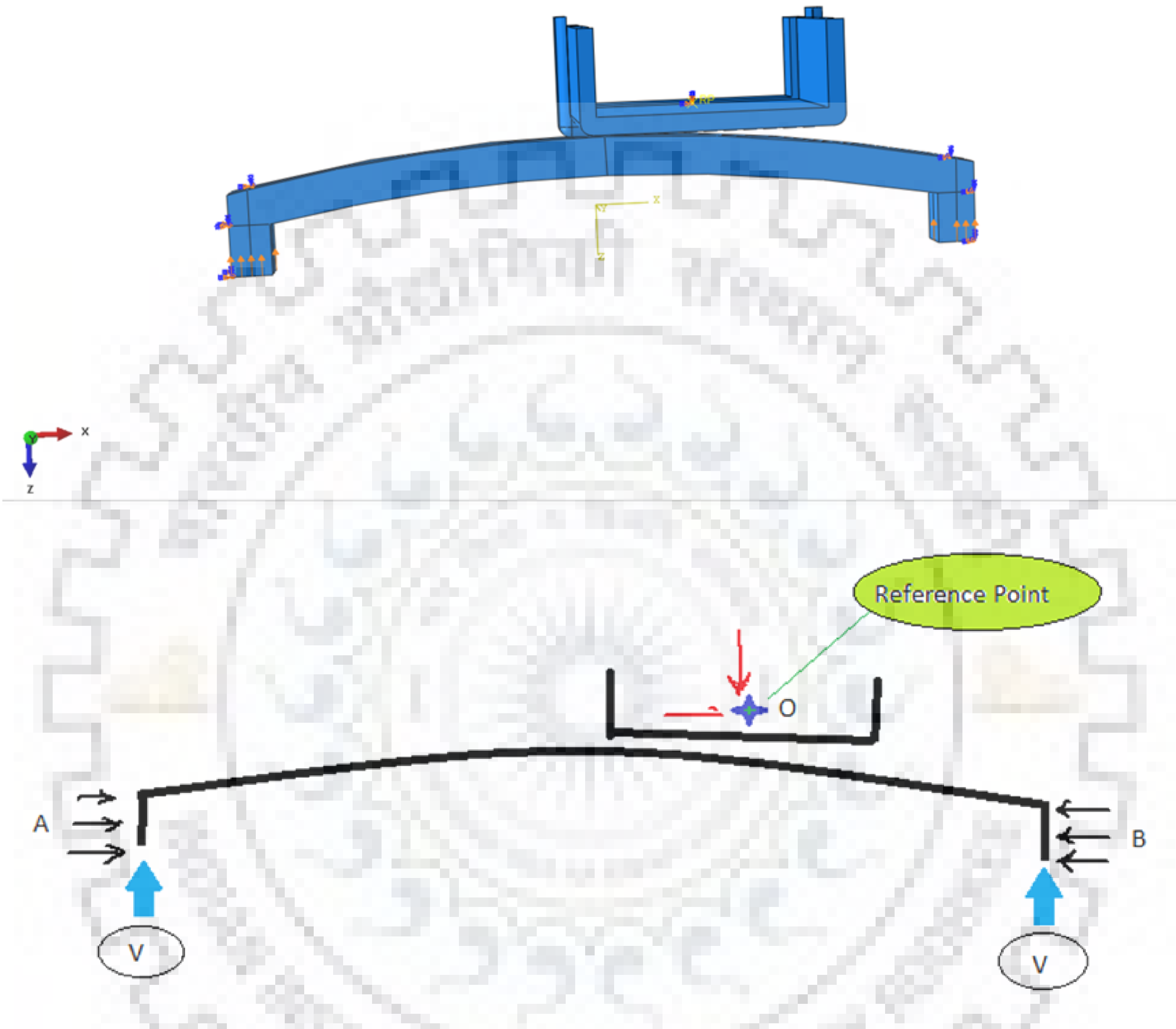

Figure 5.9: Boundary condition assignment to different nodes

At A and B point all the nodes have taken single degree of freedom as the displacement in the direction of velocity where the rotation and displacement in remaining directions are kept zero (fixed).

5.2 Processing

5.21 Job Creation:

After boundary condition job input file is created which contains all the data like boundary condition meshing detail and method of contact modelling here, I have used penalty method and tangential behaviour with friction coefficient = 0.4. Job file is submitted for deformation, stress, energy variation during contact impact.

5.3 Post processing

5.31 Results:

After processor completes its analysis and simulation work the output files are generated that can be in form of visual images, Text documents, Animations and graphs this can be viewed by using visualisation module in Abaqus. The bellow results show the displacement (deformation) plot in the direction of velocity (negative y-direction). At the start of movement of Bumper there will be zero displacement in bumper and as the contact initialises, we can see the deformation at the potential contact zone.

- **Displacement(deformation) variation:**

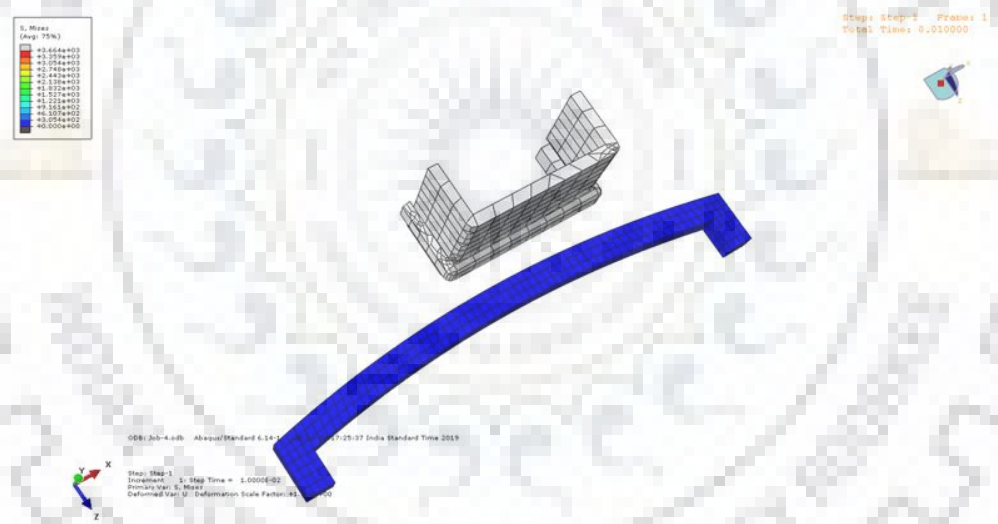

Figure 5.10: Displacement (Deformation) at t = 0 second

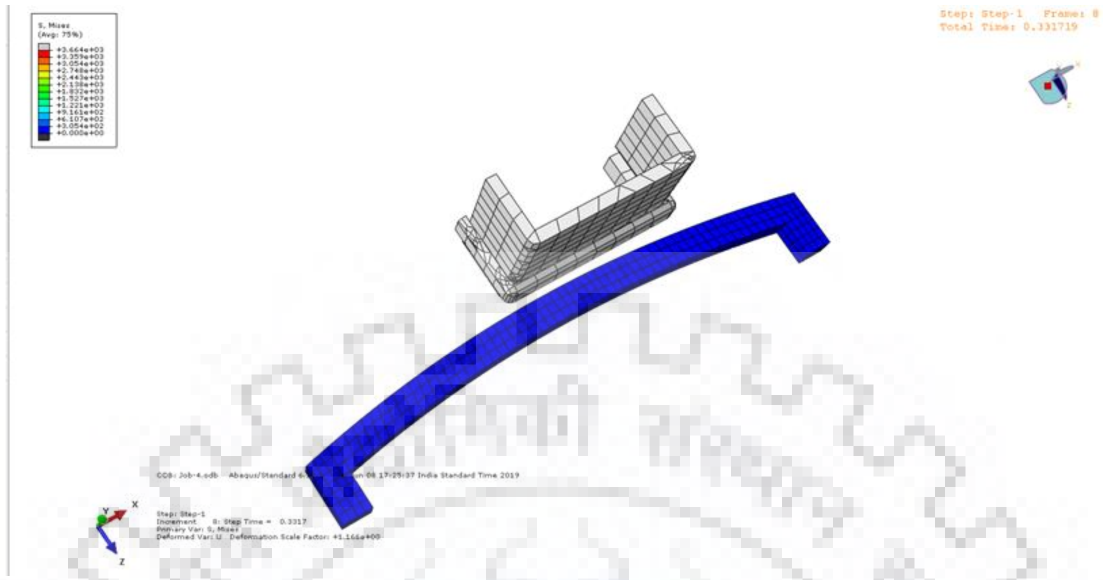

Figure 5.11: Displacement plot at t= 0

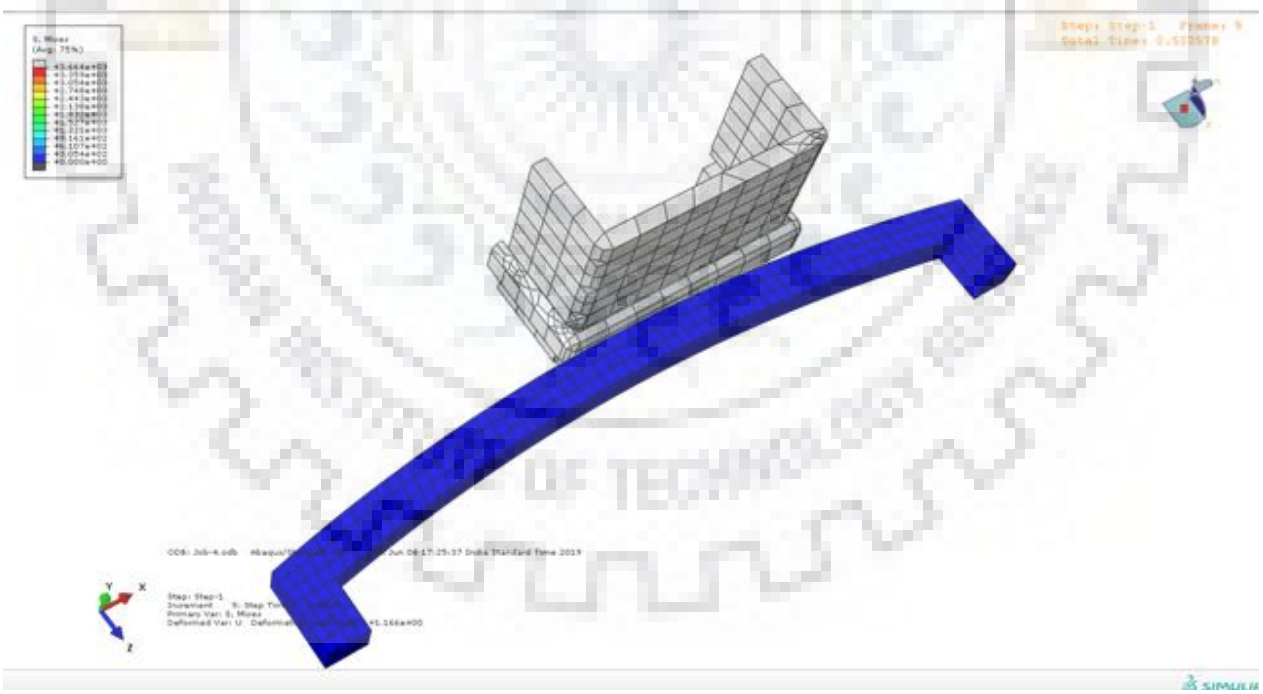

Figure 5.12: Displacement plot after t = 6 seconds

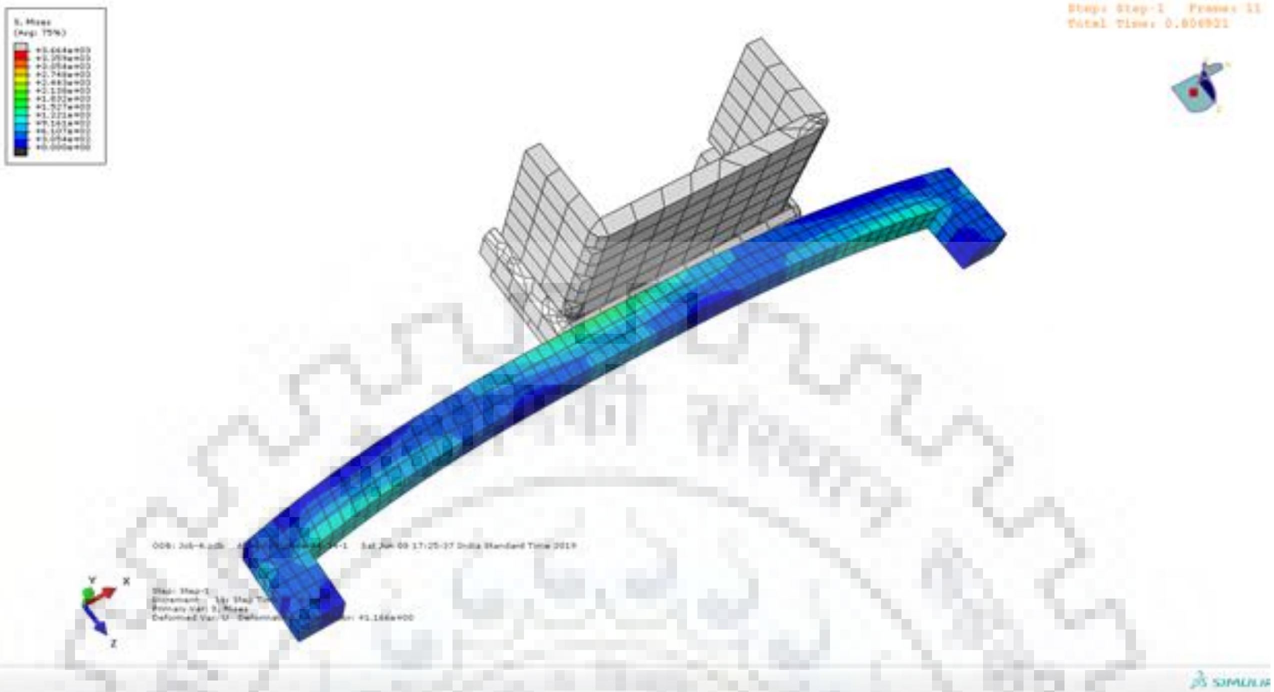

Figure 5.13: Displacement plot after t = 12 seconds

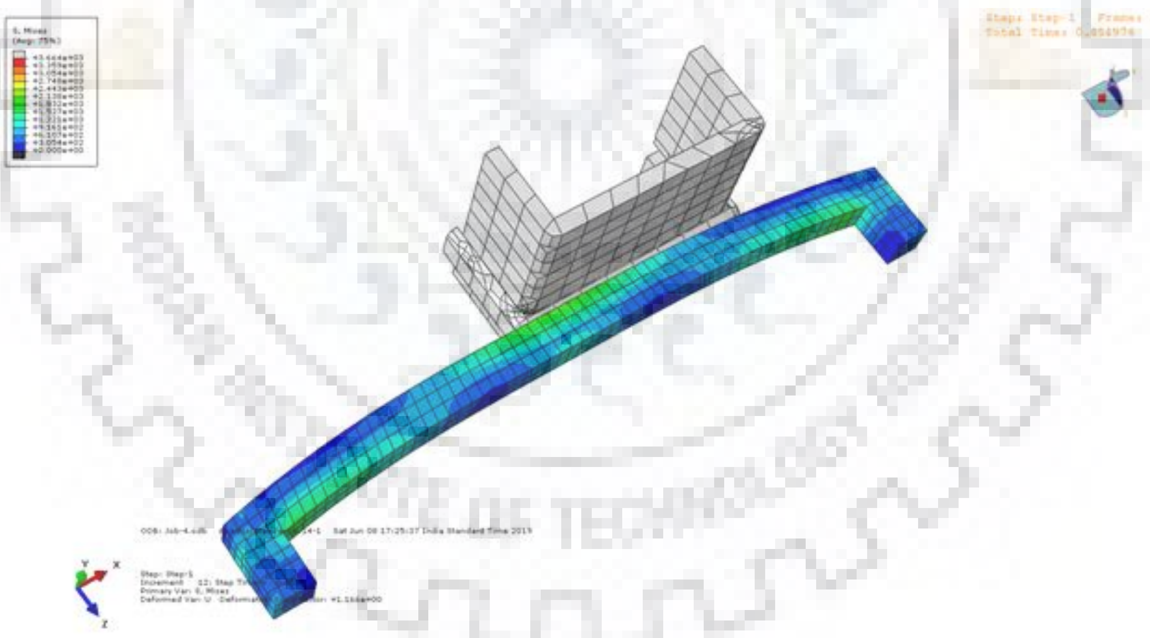

Figure 5.14: Displacement plot after t = 18 seconds

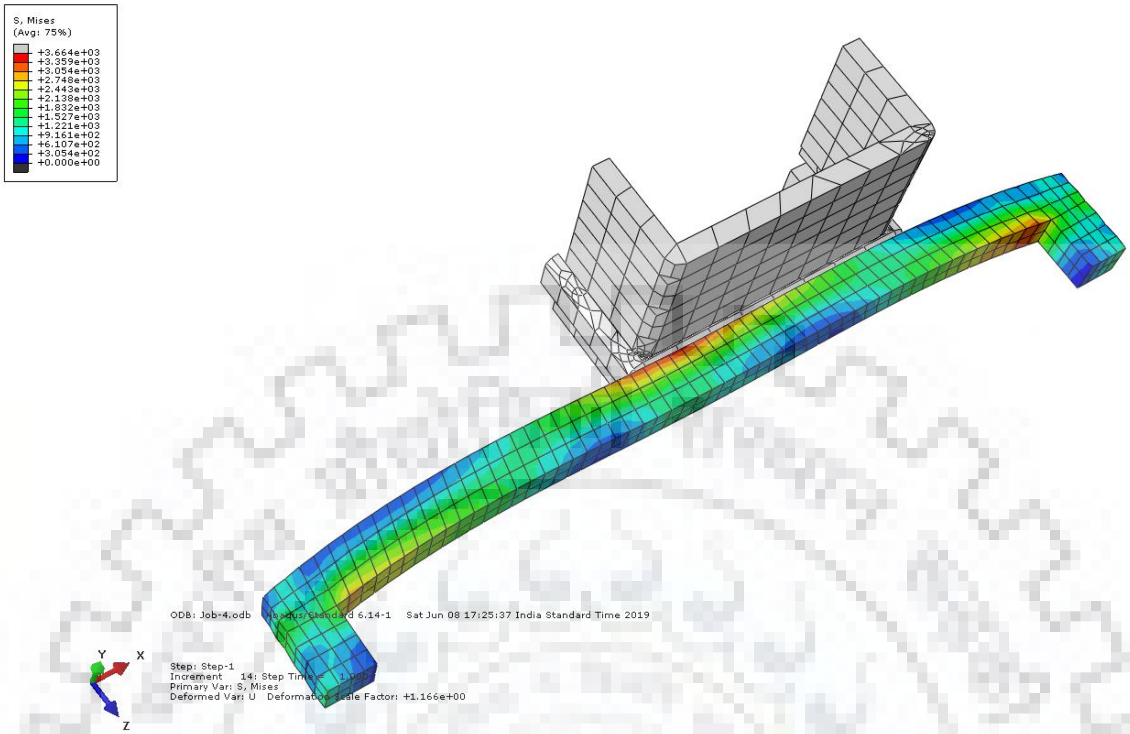

Figure 5.15: Displacement plot after t = 24 seconds

- **Stress Variation:**

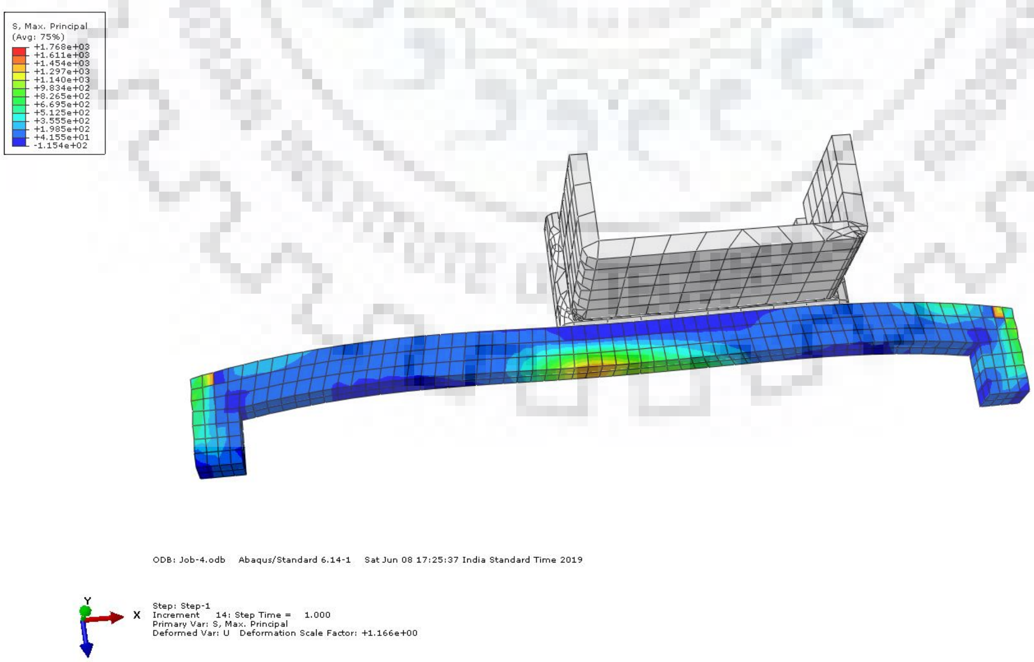

Figure 5.16: Maximum principal (in-plane) stress

After t = 24 seconds

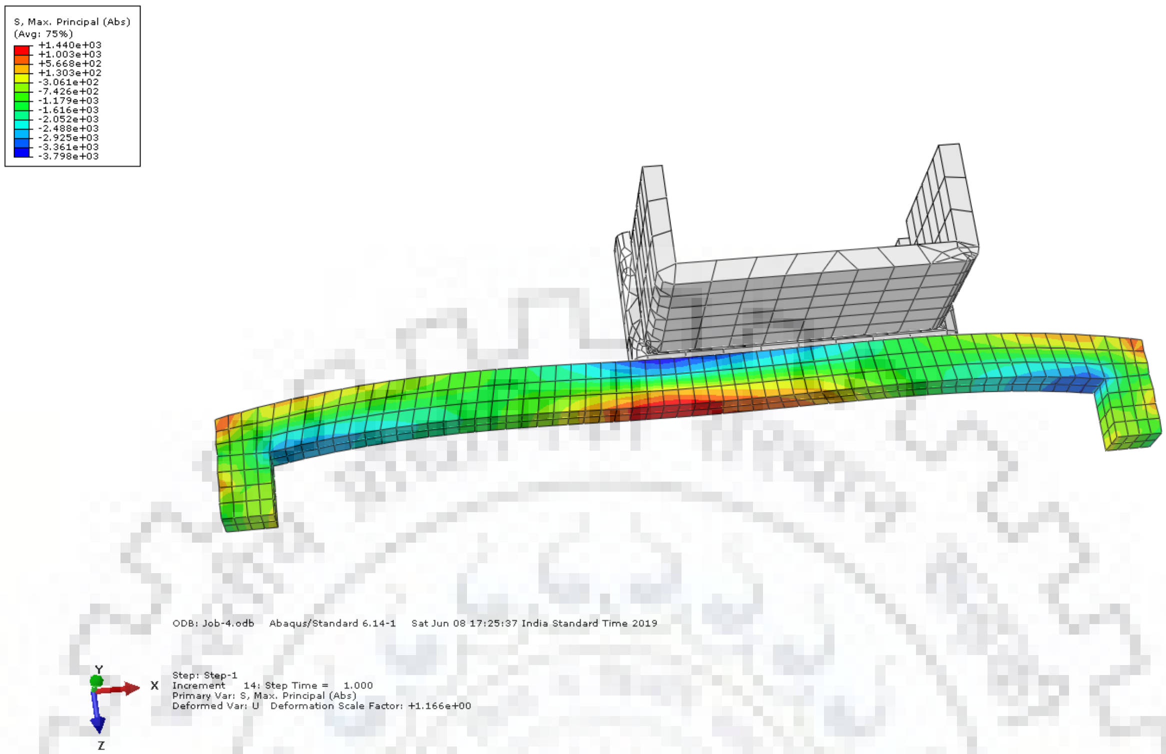

Figure 5.17: Maximum principal (Absolute) stress

After t = 24 seconds

In figure 5.13 we can that the deformation is maximum where maximum and proper contact occurs. In designing of bumper, we have to consider this maximum deformation value as the maximum strength of bumper.

According to rigidity criterion maximum deformation occurred in bumper-

$$\delta_{max} \leq \delta_{permissible}$$

We can also design the bumper based on strength criterion using maximum principal stress (Figure 5.15) as the strength of the bumper beam.

$$\sigma_{max} \leq \sigma_{permissible}$$

CHAPTER 6

ISOGEOMETRIC CONTACT MODELLING

As it was discussed above Isogeometric analysis uses NURBS geometry for both modelling and analysis, so the process like CAD modelling Material properties and boundary conditions will be included in this as well but the algorithm of process will be different because there is no automated softwares are available for Isogeometric analysis hence everything is done on MATLAB after importing the CAD geometry in MATLAB.

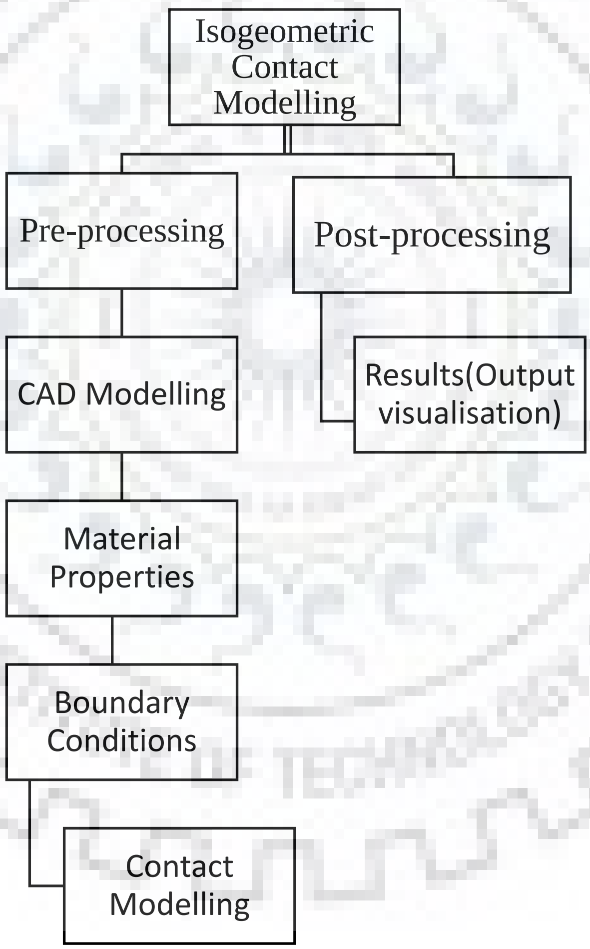

Figure 6.1: IGA simulation procedure

6.1 Pre-processing:

6.11 CAD Modelling:

Like FEM contact simulation here also CAD geometric modelling is the first step for IGA contact simulation. Geometry either can be created in MATLAB itself by using control points and knot vector or it can be imported from some NURBS modelling software. Since Bumper and pole geometry combinedly will be quite difficult to create in MATLAB, so it is generated using RHINO and saved as ***bumper.gio*** file.

Figure 6.2: Bumper – pole assembly modelled in Rhino

The imported file contains set of control point and knot vectors that will represent the geometric details.

6.12 Material Properties

Material properties are given to the geometry. Since we are doing a comparative study between IGA and FEM contact formulation, hence material properties will be same for both the simulation. A structural model supports only homogeneous isotropic materials. Therefore, all material properties must be numeric scalars. Here Steel is used as structural material Young's modulus and poisons ratio is given as

```
E0            = 2.1e5;   % Young's modulus
nu0           = 0.28;   % Poisson's ratio
```

6.13 Boundary conditions:

All the set of potential contacting knots of rigid pole are kept fixed means knots have zero displacement and rotation. The deformable bumper beam is made to move in the direction of rigid pole by providing a particular velocity.

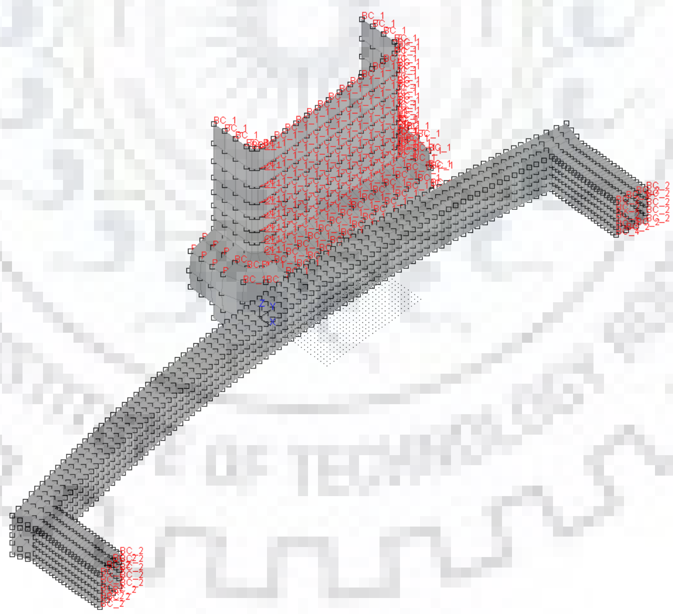

Figure 6.3: Boundary nodes as highlighted in Red

6.14 Contact modelling:

After assigning boundary conditions master and slave surfaces are made to contact each other. Here rigid pole is considered as master surface and the deformable bumper is termed as slave surface. Global search is performed and potential contact region is assigned after in local search step exact contact knots are identified. Penetration gap is made zero using iterative Newton-Raphson method.

6.2 Post-processing

6.21 Results (Output visualisation):

After contact modelling the output file is generated and saved as ***bumper.in.flavia.res*** it can be seen using **Para view Software.**

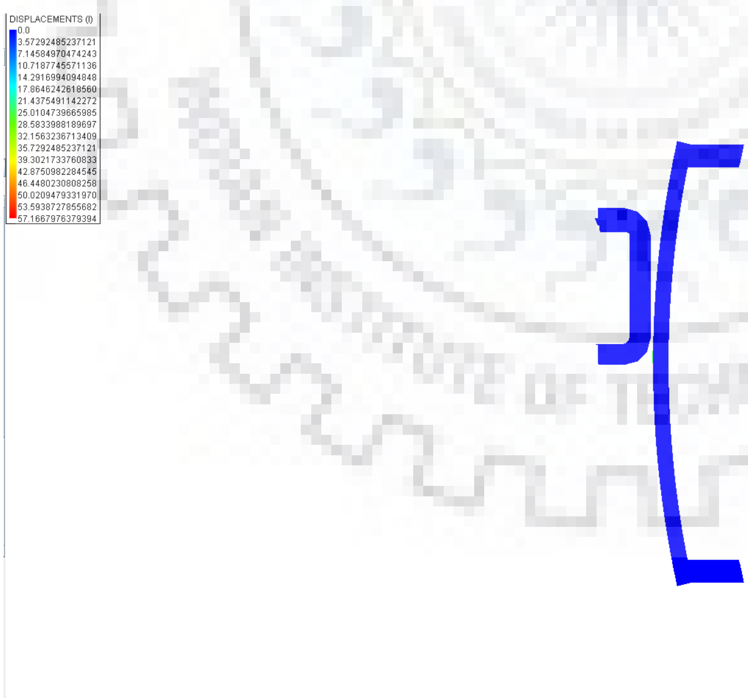

Figure 6.4: Contact initialisation at **t = 0 sec.**
Min. and max. Deformation (Displacement) = 0

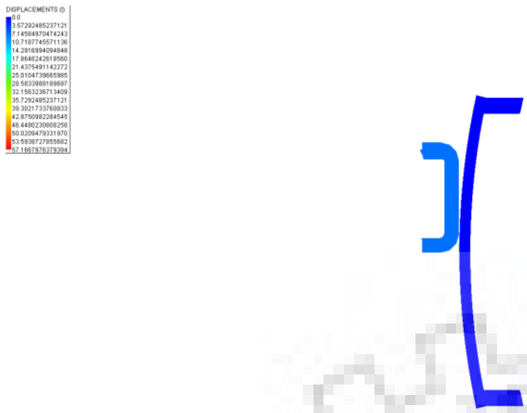

Figure 6.5:
Displacement plot after
t = 2 seconds

Figure 6.6:
Displacement plot after
t = 4 second

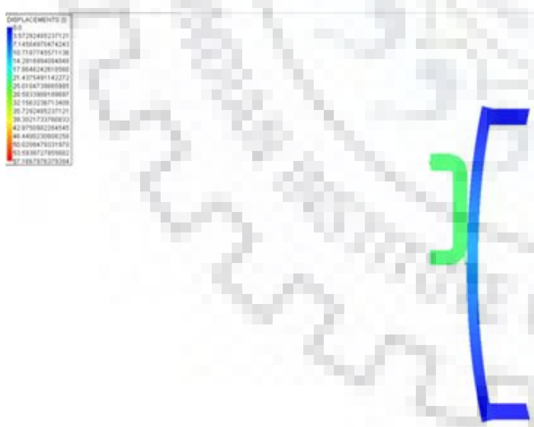

Figure 6.7:
Displacement plot after
t = 6 second

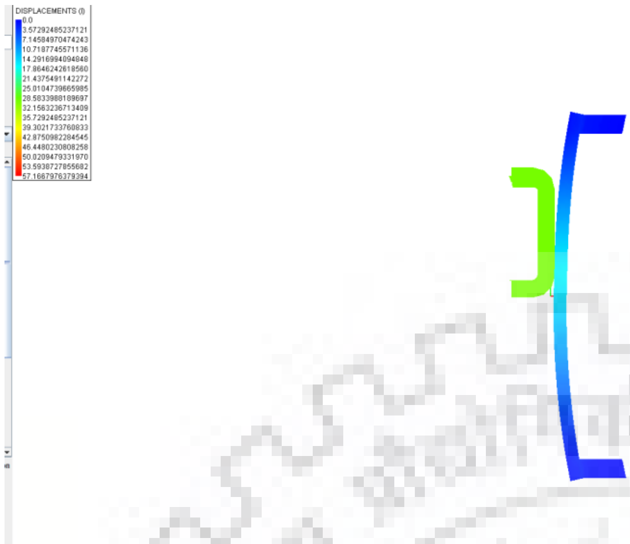

Figure 6.8:
Displacement plot after
t = 8 seconds

Figure 6.9:
Displacement plot after
t = 10 seconds

Figure 6.10:
Displacement plot after
t = 12 seconds

Figure 6.11: Displacement plot after t = 14 seconds

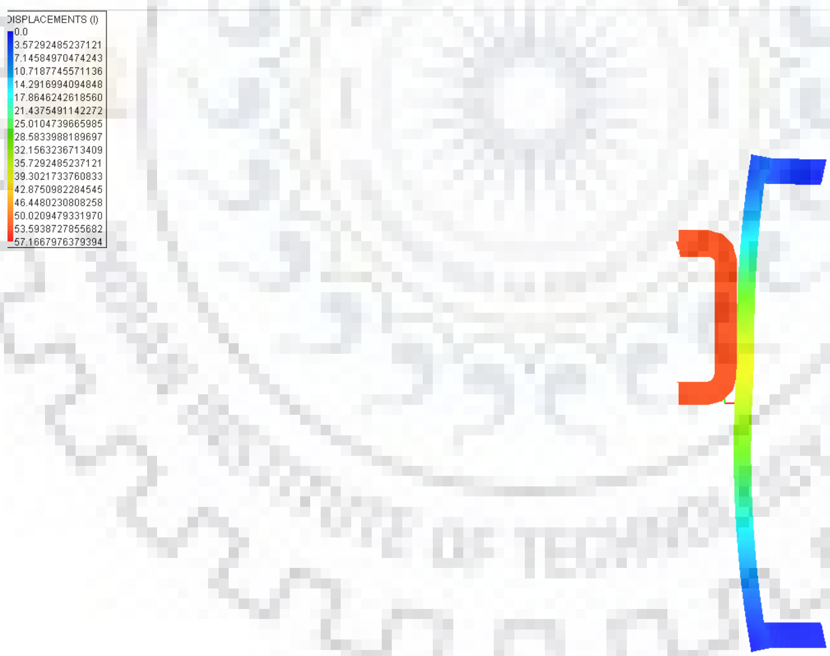

Figure 6.12: Displacement plot after t = 16 seconds

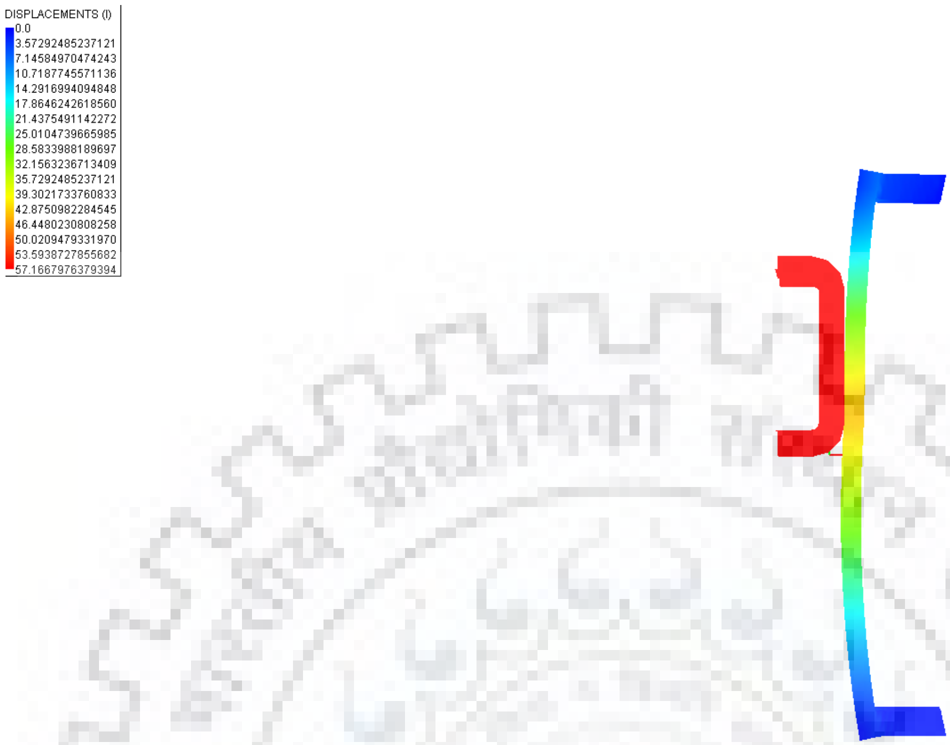

Figure 6.13: Displacement plot after t = 18 seconds

6.3 NURBS vs. FEM for contact modelling:

FEM has been used in almost every contact modelling because this process becomes easy in contact detection but due to inexact geometric discretization it is quite inefficient compared to NURBS.

NURBS has very good efficiency in every aspect and has better result as well. There are following reasons that can prove that proposed NURBS based contact modelling is better than FEM.

6.31 Meshing:

As we know meshing or discretization is the main and primary step in contact modelling and contact modelling results totally depends on that how well the model is meshed and discretized here NURBS is way better than FEM because FEM can never represent exact geometry. NURBS discretized surface is very smooth and can be taken as exact model surface.

6.32 Time Taking:

As we know contact modelling takes 30 to 40 percent total crash CPU time hence time becomes a critical constraint. The time taken by FEM modelling is greater than the NURBS modelling because FEM uses large number of elements for its better geometric representation where NURBS has flexibility for using small number of control points or knot. Due such large number of elements in FEM time taken in analysis at every node of elemental mesh becomes quite high than that of NURBS.

6.33 Results:

Due to difference in discretization NURBS and FEM has much differences in results. NURBS has exact geometric representation due to which we can trust on NURBS result than FEM. This differences in results becomes critical in some cases like contact modelling in balloon angioplasty and contact between human body and airbag.

CHAPTER 7

CONCLUSION AND FUTURE WORK

7.1 Conclusions:

As we can see the total time taken in FEM simulation is 24 seconds and total time taken in IGA contact simulation is 18 seconds so it saves 25% of overall simulation time as compared to FEM contact simulation the design of vehicle bumper is mainly depends upon how much stresses are developed during particular impact test condition so it is very important to know actual stress variation in results of virtual impact test compared to real-time impact . FEM simulation has 5.67% variation from real data and in IGA it becomes 3.47%. It looks little difference in both the simulation results but as we know this simulation is only of front vehicle bumper as we will go in full vehicle or multibody simulations it will show more variations in result data.

7.2 Future Work:

Development of interactive software platform for performing isogeometric analysis will be a big breakthrough in the field of computer aided engineering. Till now separate computational programming for each problem is done. A general computer program to perform isogeometric analysis is on the horizon and has potential to revolutionize the design and analysis industry. Automation of isogeometric analysis is a mighty aim.

REFERENCES

[1] W. G. Zhong and Z. H. Zhong: Computer Simulation of the Crashworthiness of the Energy Absorbing Front End of a Passenger Car. Automotive Engineering. Vol. 24 (2002), pp. 6-10.

[2] J. McQuaid and N. Jones, (1999) "A re-examination of Andrews research on impact resistance of railway axles," International Journal of Impact Engineering 22(7), pp. 727–738,

[3] N. Jones, (2003) "Several phenomena in structural impact and structural crashworthiness," European Journal of Mechanics/A Solids 22(5), pp. 693–707,

[4] J. A. Newman, (1980) "Head injury criteria in automotive crash testing," in Stapp Car Crash Conference. Twenty-fourth. Proceedings. Warrendale, Society of Automotive Engineers, pp. 701–747,

[5] M. M. Kamal, (1970) "Analysis and Simulation of Vehicle to Barrier Impact," SAE Technical Paper 700414,

[6] O. Zienkiewicz, (1977) "The finite element method", McGraw-Hill.

[7] J.A. Bennett, R.V. Lust, and J.T. Wang, (1991) "Optimal design strategies in crashworthiness and occupant protection, in Crashworthiness and Occupant Protection in Transportation Systems", AM D-Vol. 126/BED-Vol. 19, ASME, New York, pp. 51-66.

[8] Yehia A. Abdel-Nasser, (2003) "Frontal crash simulation of vehicles against lighting columns using FEM" Naval Architecture and Marine Engineering Department, Faculty of Engineering, Alexandria University, Egypt

[9] Joshi, K. (2012). CAE Process Automation – Crash Simulation.

[10] R. P. R. Cardoso & O. B. Adetoro (2017) "On contact modelling in isogeometric analysis", European Journal of Computational Mechanics, 26:5-6, 443-472, DOI: 10.1080/1777179.2017.1354575

[11] Nguyen et al. "Isogeometric analysis: An overview and computer implementation aspects", Mathematics and Computers in Simulation 117 89–116

[12] Song Wang et al. (2000) "Contact search strategies for FEM simulation of the blow molding process", International journal for numerical methods in engineering Int. J. Numer. Meth. Engng; 48(4):501-521

[13] Vladislav A. Yastrebov,(2011) "Computational Contact Mechanics,geometry detection and numerical techniques" Materials. Ecole Nationals Superieure des Mines de Paris 23:43-57

[14] G J Gao & H Q Tian (2007) "Train's crashworthiness design and collision analysis", International Journal of Crashworthiness, 12:1, 21-28, DOI: 10.1533/ijcr.2006.0138

[15] Ni, C. M., and J. O. Song. (1986) "Computer-aided design analysis methods for vehicle structural crashworthiness." Proc. of Symposium on Vehicle Crashworthiness Including Impact Biomechanics AMD. Vol. 79.

[16] Jonsén, Pär, et al. (2009) "Identification of lumped parameter automotive crash models for bumper system development." International Journal of Crashworthiness 14.6: 533-541.

[17] P.D. Bios, "Vehicle Crashworthiness and Occupant Protection in Transportation System", book.

[18] Wen, Zefeng, XuesongJin, and Weihua Zhang. (2005) "Contact-impact stress analysis of rail joint region using the dynamic finite element method." *Wear* 258.7-8 :1301-1309.

[19] Santos, Daniel Barbedo Vasconcelos, and Alex Alves Bandeira. (2018) "Numerical modeling of contact problems with the finite element method utilizing a B-Spline surface for contact surface smoothing." Latin American Journal of Solids and Structures15.8 .

[20] Temizer, I., P. Wriggers, and T. J. R. Hughes. (2011) "Contact treatment in isogeometric analysis with NURBS." Computer Methods in Applied Mechanics and Engineering 200.9-12: 1100-1112.

[21] Yang, Y.-B. and Kuo, S.-R. (1994). Theory and analysis of nonlinear framed structures.

[22] Spink, M., Claxton, D., de Falco, C., and V ́azquez, R. (2014). Nurbs toolbox. Matlab Central [online], Accessed on June, 15.

[23] Kim, C. and Arora, J. (2003). Nonlinear dynamic system identification for automotive crash using optimization: A review. Structural and Multidisciplinary Optimization, 25(1):2–18.

[24] Temizer et. al. (2011) 'A large deformation frictional contact formulation using NURBS-based isogeometric analysis' INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN ENGINEERING Meth. Engng; 87:1278–1300

[25] Karthikeyan, M., Jenarthanan, M.P., Giridharan, R. et al. Trans Indian Inst Met (2019). https://doi.org/10.1007/s12666-019-01741.

[26] Fujun, W., Jiangang, C. & Zhenhan, Y. Acta Mech Sinica (2000) 16: 374. https://doi.org/10.1007/BF02487690

[27] Cottrell, J. A., Hughes, T. J., and Bazilevs, Y. (2009). Isogeometric analysis: toward integration of CAD and FEA. John Wiley & Sons.